

Lecture Notes in Mechanical Engineering

Arun K. Saha

Debopam Das

Rajesh Srivastava

P.K. Panigrahi

K. Muralidhar *Editors*

Fluid Mechanics and Fluid Power — Contemporary Research

Proceedings of the 5th International
and 41st National Conference on FMFP 2014

 Springer

Lecture Notes in Mechanical Engineering

About this Series

Lecture Notes in Mechanical Engineering (LNME) publishes the latest developments in Mechanical Engineering—quickly, informally and with high quality. Original research reported in proceedings and post-proceedings represents the core of LNME. Also considered for publication are monographs, contributed volumes and lecture notes of exceptionally high quality and interest. Volumes published in LNME embrace all aspects, subfields and new challenges of mechanical engineering. Topics in the series include:

- Engineering Design
- Machinery and Machine Elements
- Mechanical Structures and Stress Analysis
- Automotive Engineering
- Engine Technology
- Aerospace Technology and Astronautics
- Nanotechnology and Microengineering
- Control, Robotics, Mechatronics
- MEMS
- Theoretical and Applied Mechanics
- Dynamical Systems, Control
- Fluid Mechanics
- Engineering Thermodynamics, Heat and Mass Transfer
- Manufacturing
- Precision Engineering, Instrumentation, Measurement
- Materials Engineering
- Tribology and Surface Technology

More information about this series at <http://www.springer.com/series/11236>

Arun K. Saha · Debopam Das
Rajesh Srivastava · P.K. Panigrahi
K. Muralidhar
Editors

Fluid Mechanics and Fluid Power – Contemporary Research

Proceedings of the 5th International
and 41st National Conference
on FMFP 2014

Editors

Arun K. Saha
Department of Mechanical Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

P.K. Panigrahi
Department of Mechanical Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

Debopam Das
Department of Aerospace Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

K. Muralidhar
Department of Mechanical Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

Rajesh Srivastava
Department of Civil Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

ISSN 2195-4356 ISSN 2195-4364 (electronic)
Lecture Notes in Mechanical Engineering
ISBN 978-81-322-2741-0 ISBN 978-81-322-2743-4 (eBook)
DOI 10.1007/978-81-322-2743-4

Library of Congress Control Number: 2016931198

© Springer India 2017

This work is subject to copyright. All rights are reserved by the Publisher, whether the whole or part of the material is concerned, specifically the rights of translation, reprinting, reuse of illustrations, recitation, broadcasting, reproduction on microfilms or in any other physical way, and transmission or information storage and retrieval, electronic adaptation, computer software, or by similar or dissimilar methodology now known or hereafter developed.

The use of general descriptive names, registered names, trademarks, service marks, etc. in this publication does not imply, even in the absence of a specific statement, that such names are exempt from the relevant protective laws and regulations and therefore free for general use.

The publisher, the authors and the editors are safe to assume that the advice and information in this book are believed to be true and accurate at the date of publication. Neither the publisher nor the authors or the editors give a warranty, express or implied, with respect to the material contained herein or for any errors or omissions that may have been made.

Printed on acid-free paper

This Springer imprint is published by SpringerNature
The registered company is Springer (India) Pvt. Ltd.

Preface

The 5th International and 41st National Conference on Fluid Mechanics and Fluid Power was held in the campus of Indian Institute of Technology Kanpur during December 12–14, 2014. A total of 450 scientific participants, 275 of whom were students, attended the conference. The participants had fruitful scientific and technical discussions and exchanges that made the conference a complete success.

The overall response to the conference was quite encouraging. A large numbers of papers were received. After careful review, 397 papers were chosen for the inclusion for presentation during the conference. Among these, 286 papers were chosen for oral presentation while the rest were displayed in the form of posters. Subsequent to conference, an additional round of review was carried out. Among the presented papers, 161 papers were selected for inclusion in the published proceedings.

The papers appearing in the proceedings are distributed into 14 major parts:

1. Fundamental Issues and Perspective in Fluid Mechanics
2. Measurement Techniques and Instrumentation
3. Computational Fluid Dynamics
4. Instability, Transition, and Turbulence
5. Turbomachinery
6. Multiphase Flows
7. Fluid-structure Interaction and Flow-induced Noise
8. Microfluidics
9. Bio-inspired Fluid Mechanics
10. IC Engines and Gas Turbines
11. Transport Phenomena in Materials Processing and Manufacturing
12. MHD and EHD Flows
13. Granular Flows
14. Nuclear Reactor Thermal Hydraulics

These papers represent the most recent research on the subject. The editors would like to thank all the authors and the anonymous referees for paying attention

to the quality of the publications. Partial financial support for publishing the conference proceedings is due to the financial support received from IITK, ISRO, DST, DRDO, and DAE and companies, including Coolflo (Hyderabad), GE Global Research (Bangalore), TESSCORN (Bangalore), Laser Science Services (Mumbai), and TSI Instruments (Bangalore).

The contents of these proceedings reveal the breadth of current activities in different themes related to fluid mechanics. We hope they form a useful starting point for beginners as well as practitioners in this discipline.

Kanpur, India

Arun K. Saha
Debopam Das
Rajesh Srivastava
P.K. Panigrahi
K. Muralidhar

About the Conference

Over the years, the country has made substantial progress in the fields related to fluid mechanics and fluid power. This development can be attributed to rapid advancements in computational methods and experimental techniques. The outcome has immensely benefitted the industry in numerous ways. In order to disseminate research findings and academic contributions relevant to the industry, the National Society of Fluid Mechanics and Fluid Power (<http://www.me.iitb.ac.in/~fmfp/>) holds a national conference every year and international conference once in every 4 years, either at an academic institution or at a research organization.

The National Society of Fluid Mechanics and Fluid Power was established in 1973 at IIT Bombay as a professional society to promote activities in the fields of fluid mechanics and fluid power in India. The annual conference has been held (mostly) uninterrupted over the years and the one in December 2014 is the 5th International and 41st National in the series. The society provides a common platform for academicians, engineers, and technologists to come together and discuss progress made within the country and abroad on various aspects of fluid mechanics and fluid power.

A total of 750 papers were received in the International FMFP Conference, 2014 and after a proper review process, 397 papers were selected for technical presentation—both oral and poster. A plenary lecture and several keynote lectures were scheduled during the two and half days of the conference. Over 450 college faculty, scientists, engineers, and students participated in the conference in the cool and quite ambience of Indian Institute of Technology Kanpur (India).

We acknowledge the effort of various committees, dedicated staff and student volunteers who contributed towards the successful completion of the conference. The success of the conference is also due to the generous financial support received from organizations such as IITK, ISRO, DST, DRDO, and DAE and companies including Coolflo (Hyderabad), GE Global Research (Bangalore), TESSCORN (Bangalore), Laser Science Services (Mumbai), and TSI Instruments (Bangalore).

Contents

Set 1

Part I Fundamental Issues and Perspective in Fluid Mechanics

Effect of Inlet Shear on the Flow Structures Associated with Elevated Jet in Cross-Flow at Low Reynolds Number	3
Sachidananda Behera and Arun K. Saha	
Slip Flow of a Shear-Thinning Power-Law Fluid Past an Assemblage of Spherical Particles	11
Rahul R. Ramteke and Nanda Kishore	
Design and Characterization of Liquid Centered Swirl-Coaxial Injector	23
Kamalakkannan Kannaiyan and Aravind Vaidyanathan	
Wall Shear Rates Generated During Coalescence of Pendant and Sessile Drops	33
Praveen M. Somwanshi, K. Muralidhar and Sameer Khandekar	
Effect of Aft Wall Offset on Supersonic Flow over Cavity	43
Praveen Kumar Maurya, C. Rajeev, R.R. Vinil Kumar and Aravind Vaidyanathan	
Effect of Free Surface on Submerged Synthetic Jet Parallel to the Surface	53
Abhay Kumar, Manish Gupta, Arun K. Saha and P.K. Panigrahi	
Study of Shock Wave/Turbulent Boundary Layer Interaction over a Forward-Facing Step	63
Jayaprakash N. Murugan and Raghuraman N. Govardhan	
Energy Loss Minimizing Drag Reduction and Self-propulsion Strategies	73
Pritam Giri and Ratnesh K. Shukla	

Design of a Cyclone Separator for Cleaning of Dust from Volumetric Air Receiver	83
Gurveer Singh, Dheeraj Saini, Laltu Chandra and Rajiv Shekhar	
Influence of Delta Wing Vortex Generator on Counter Rotating Vortex Pair in Film Cooling Application of Gas Turbine Blade.	95
Nirmal Halder, Arun K. Saha and P.K. Panigrahi	
Influence of Aeration on the Hydrodynamic Behavior of a Pressurized Circulating Fluidized Bed.	105
Azd Zayoud, Yerbol Sarbassov, P. Mahanta, U.K. Saha and Sai Gu	
Characteristics of Two-Dimensional Flow Past a Square Cylinder Placed in a Channel with High Blockage	115
Abhishek Verma, Arun K. Saha and Sachidananda Behera	
Aerothermodynamics of a Blunt Airfoil with a Flow-Through Duct for Hypersonic Flight	125
Rajesh Yadav, Mohammad Faisal and Dheeraj Minglani	
Rayleigh-Taylor Instability Induced Liquid Atomization.	135
M. Vadivukkarasan and Mahesh V. Panchagnula	
Onsager's Principle Consistent 13 Moments Transport Equations	145
N. Singh and Amit Agrawal	
 Part II Measurement Techniques and Instrumentation	
Experimental Investigations on Onset of Slugging in Horizontal Air-Water Two-Phase Flow	157
Jignesh Thaker and Jyotirmay Banerjee	
Study of Orifice Inclination Effect on Synthetic Jet Characteristics Using Laser Doppler Velocimetry and Laser Induced Fluorescence	167
B.P. Maddilety, Arun K. Saha and P.K. Panigrahi	
Particle Image Velocimetry Study of Sonic Jet Injected into Supersonic Crossflow	177
Nithiyaraj Munuswamy and Raghuraman N. Govardhan	
Investigations into Asymmetric Oscillations of a Symmetric Airfoil	185
Anshul Khandelwal, Kamal Poddar and Debopam Das	
Exhaust Gas Flow Field Simulation of an Internal Combustion Engine for a Thermal Sensor.	195
Sumit Agarwal and Niranjana Sahoo	
Estimation of Equilibrium Moisture Content and Drying Time of Potato Through Hot Air Drying	205
V.P. Chandramohan and Prabal Talukdar	

Effect of Sodium Pyrophosphate and Polyethylene Oxide on Thermal Diffusivity of Laponite RD Colloidal Suspension. 215
 Susheel S. Bhandari

Numerical Simulation and Experimental Validation of Future FBR Surge Tank Hydraulics 229
 P. Lijukrishnan, D. Ramdasu, V. Vinod, G. Padmakumar and K.K. Rajan

Flow Characteristics of Synthetic Jet on Torpedo Shape Model in Cross Flow. 239
 Abhay Kumar, Manish Gupta, Arun K. Saha and P.K. Panigrahi

Part III Computational Fluid Dynamics

CFD Simulation Studies of High Performance Computing (HPC) Facilities 251
 Ajinkya N. Deshmukh, Rahul S. Pawar, Sourabh S. Kulkarni, Vikas Kumar, Sandeep K. Joshi and Mangesh B. Chaudhari

Effect of Wall Proximity on the Flow Over a Heated Square Bluff Body 269
 Deepak Kumar and Amit Dhiman

A Finite Volume Moving Mesh Method for the Simulation of Compressible Flows. 279
 Ratnesh K. Shukla and Harshavardhana S. Pathak

Reverse Flow in Converging Channels with Obstruction at the Entry 289
 Senthil Prasad Mathiyalagan and B.H.L. Gowda

A Hybrid Aerodynamic Shape Optimization Approach for Axisymmetric Body in Hypersonic Flow 301
 Shuvayan Brahmachary, Ganesh Natarajan and Niranjan Sahoo

Control of Vortex Shedding Behind a Circular Cylinder Using a Combination of Slot and Control Plates. 313
 S. Karthikeyan and S. Senthilkumar

Improved Methodology for Mass Conservation in Sharp Interface Immersed Boundary Method for Moving Boundaries 323
 Manish Kumar and Somnath Roy

Computation of the Flowfield in the Vicinity of an Electric Vehicle Platform 333
 K. Panchal and M. Damodaran

Implementation of Matrix Dissipation Scheme for Numerical Simulations of Two Dimensional Navier-Stokes Equations. 343
 Shikhar Jaiswal and J.S. Mathur

Effect of Reynolds Number, Strouhal Number and Amplitude of Oscillation on Pulsating Flow Characteristics Through a Circular Pipe	353
Vedant U. Chavan and C.M. Sewatkar	
Mixed Convection in a Ventilated Enclosure with Different Heater Position	363
Nirmalendu Biswas, Pallab Sinha Mahapatra and Nirmal K. Manna	
Evaluation of Flow Properties of Air at the Exit of Holes on the Blow Pipe in a Pulse Jet Filter Bag House	375
Jayaram Thumbe, Ravikiran Kadoli and K.V. Gangadharan	
Forced Convection from an Inclined Elliptical Cylinder with Constant Heat Flux: Effect of Prandtl Number	385
S.A. Patel and R.P. Chhabra	
Laminar Natural Convection in a Square Duct from Two Heated Horizontal Cylinders: Effect of Prandtl Number	395
A.K. Baranwal and R.P. Chhabra	
Flow Analysis of Subsonic and Supersonic Nozzles	405
Dharti Parmar, Meha Setiya, Beena Baloni and S.A. Channiwala	
A Comparative Analysis of Heat Transfer in Extended Surfaces with and Without Holes	421
Saroj Yadav, Koushik Das and Krishna Murari Pandey	
Computation of Variable Density Flows on Hybrid Unstructured Grids	431
Ankit Bajpai, Jai Manik, Mukul Parmananda, Amaresh Dalal and Ganesh Natarajan	
Numerical Simulation of Solidification and Melting Problems on Unstructured Grid	439
Debraj Das and Amaresh Dalal	
Numerical Simulation of Pulsatile Flow Past Heated Square Cylinders	449
Mohammad Khalid, Raunaq Haseeb and Syed Fahad Anwer	
3D Unsteady Numerical Simulation of All-Vanadium Redox Flow Battery	457
H.M. Sathisha and Amaresh Dalal	
Computation of Flow Coupled with the Electric Field on Unstructured Grid	467
G.N. Sunil, Amaresh Dalal and Ganesh Natarajan	

A Combined Implementation of Overset Grid and Patched Grid Methodologies	477
Jyothi Kumar Puttam	
Numerical Investigation of Free Jet Expansion of Gas Mixture	487
Pranit Gaikwad, Arvind Deshpande, Atul Tak and A.K. Das	
CFD Study on Thermal Hydraulic Performance of A Wavy Channel Based PCHE Model	497
Hamid Hassan Khan, A.M. Aneesh, Atul Sharma, Atul Srivastava and Paritosh Chaudhuri	
Thermo-Hydraulic Performance of Zigzag, Wavy, and Serpentine Channel Based PCHEs	507
A.M. Aneesh, Atul Sharma, Atul Srivastava and Paritosh Chaudhuri	
CFD Analysis of Water Cooled Flame Deflector in Rocket Engine Test Facility	517
Anant Singhal, T. John Tharakan and Raju P. Thomas	
Computational Study of Three Dimensional Wall Jet on Concave Surface	529
S.V.H. Nagendra, D.V.S. Bhagavanulu and Prasant Nanda	
Design and Simulation of a Low Speed Wind Tunnel with Analysis of Wind Effects on an Airfoil (CFD)	539
Rahul Goswami, Srijan Singhal and Vijay Kumar	
A 3D Numerical Study on Opposed Flow Flame-Spread over Thin Parallel Fuel Sheets of Finite Widths in Microgravity	551
M.N. Kiran Kumar and Amit Kumar	
Simulation of Mixed Convection Flow in Square Cavities by Lattice Boltzmann Method	559
Dhrubajyoti Kashyap and Anoop K. Dass	
Unified Solver for Thermobuoyant Flows on Unstructured Meshes	569
Mukesh Kumar and Ganesh Natarajan	
 Part IV Instability, Transition and Turbulence	
Large Eddy Simulation of Smoke Spread in a Multi-storied Building	583
Srisai Meher, Gulab A. Khedekar, Vikas Kumar and Supriyo Paul	
Numerical Simulation of Isothermal Cruciform Jet Flow	595
B.T. Kannan and Sundararaj Senthilkumar	

Analysis of Turbulence in Entrance Regime of Rectangular Duct Using Hot Film Anemometer	605
Sagar M. Kadam, Jignesh P. Thaker and Jyotirmay Banerjee	
Evaluation of the Gas Kinetic Scheme Based on Analytical Solution to the BGK Model for Simulating Compressible Turbulent Flows	615
Nishant Parashar, Balaji Srinivasan and Sawan Suman	
Redesigning of Motorcycle Helmet for Improved Air Ventilation Using Numerical Simulations	625
B.S. Shishodia, S. Sanghi and P. Mahajan	
Experimental Investigation of Heat Transfer Enhancement in a Stationary Two-Pass Square Duct with V-ribs	637
Ramesh Erelli and Arun K. Saha	
Pressure Hessian Evolution in Compressible Turbulence.	649
Mohammad Danish, Sawan Suman and Balaji Srinivasan	
Self-sustained Oscillation for a Three Dimensional Transonic Cavity Using LES	659
K.M. Nair, S. Sarkar and Z. Labana	
CFD Analysis of Particle Laden Flow in a Cyclone Separator Using RANS and LES Methodologies.	669
Pratik Makwana and Absar Lakdawala	
Peculiar Behavior of Thermally Stratified Channel Flow Under Non-Boussinesq Condition.	679
S.M. Yahya, S.F. Anwer, S. Sanghi and N. Hasan	
Flow Control in T-Junction Using CFD Based Optimization	687
K. Srinivasan, V. Balamurugan and S. Jayanti	
Grid Size Optimization for Diesel Injection Spray Nozzle Using CFD Analysis	697
Balamurugan Rathinam, Duraisamy Subramanian, Upendra Naithani, Frederic Ravet and Laurent Duffoy	
Simulation of Unsteady Wall-Jet in a Confined Geometry and Identification of Coherent Structures Using Proper Orthogonal Decomposition	713
Arun Govind Neelan, A.C. Mandal and Ashoke De	
Two Phase Flow Stability Analysis of Multiple Horizontal Uniformly Heated Channel	723
Ashish Mani Mishra, Subhanker Paul, Vikas Pandey and Suneet Singh	

Experimental Study of Boundary Effects on Buoyant Jet Dynamics in Linearly Stratified Medium 737
 N. Mirajkar Harish, Alan John Maniamkot and Sridhar Balasubramanian

Part V Turbomachinery

Real-Time Pole Placement Control of the Rugged Electrohydraulic System 749
 Bikash Kumar Sarkar, Rana Saha, Saikat Mookherjee and Dipankar Sanyal

Performance Evaluation of Hydraulic Francis Runner at Different Blade Stagger 765
 Vishnu Prasad, Ruchi Khare and Arun Kumar Sharma

Detailed Heat Transfer Investigation Inside a Rectangular Duct with an Array of Ventilated Rib Turbulators 775
 Naveen Sharma, Andallib Tariq and Manish Mishra

A Novel Design Philosophy for Mixed Flow Compressor. 785
 Subhash Padmanabhan and Jyotirmay Banerjee

Effect of Slanted Perforation in Tabs for Subsonic and Transonic Jets 795
 R. Asad Ahmed, S. Thanigaiarasu, D.L.V. Vasu Kannah, S. Elangovan and E. Rathakrishnan

Computational Analysis of Unsteady Flow in Turbine Part of Turbocharger 811
 H.K. Srinivas Rao, S. Raviteja and G.N. Kumar

Computational Study of Pressure Side Film Cooling—Effect of Density Ratio with Combination of Holes. 821
 D. Radheesh, P. Ssheshan, N. Gnanasekaran and R.K. Panda

Influence of Endwall Clearance on HSV and Passage Flow Between Two Turbine Cascade Blades. 833
 V. Vikram, B.H.L. Gowda and B.B.S.S.S. Prasad

Aerothermal Analysis of Insulation to Protect Exhaust Ducts of a Pusher Turbo Prop Engine Aircraft 843
 C.A. Vinay and Bhaskar Chakravarthy

Study of Noise and Vibration Signal for a Radial Flow Pump During Performance Test 853
 S. Christopher and S. Kumaraswamy

Development of One of the World's Largest Single Stage Vertical Centrifugal Pumps for Lift Irrigation Scheme	863
V. Shrinivas Rao, Sachin Kumar Tripathi and Saurabh Sharma	
Numerical Analysis of Effects of Turbine Blade Tip Shape on Secondary Losses	871
R.S. Sarath, R. Ajith Kumar, B.V.S.S.S. Prasad and A.R. Srikrishnan	
Experimental Investigations on Pump Running in Turbine Mode with Different Draft Tubes	881
Nimit Patel, Maulik Chauhan, Sanjay V. Jain and Rajesh N. Patel	
Numerical Simulation of Regenerative Pump	891
Rahul Verma, Shyam Shukla, Ravi Birajdar and Nirmal Tiwari	
Multistage Hydraulic Cylinder Buckling Analysis by Classical and Numerical Methods with Different Mounting Conditions	901
V. Ramasamy and A.M. Junaid Basha	
CFD Analysis of Impeller-Diffuser Interaction in a Centrifugal Compressor with Twisted Vaned Diffuser.	911
P. Venkateswara Rao, G.V. Ramana Murty and G. Venkata Rao	
Numerical Studies on the Effect of Purge Flow on the Performance and Flow Field of a Gas Turbine with Disc Cavity	923
B.S. Vinay Ballal, Q.H. Nagpurwala and M.D. Deshpande	
Computational Studies on the Effect of Speed Ratio and Stagger Angle in a Counter Rotating Turbine with Respect to Flow Field and Performance	933
Rayapati Subbarao and M. Govardhan	
Set 2	
Part VI Multiphase Flows	
Petroleum Reservoir Simulation of Two-Phase Flow	947
Anugrah Singh, N. Manjunath Reddy and Pankaj Tiwari	
Collapse of a Taylor Bubble at Free Surface: An Experimental Investigation	957
Basanta K. Rana, Lukeshwar S. Paikara, Arup K. Das and Prasanta Kumar Das	
Impingement of Aviation Fuel Drop on Stainless Steel Surface	969
S. Sen, V. Vaikuntanathan and D. Sivakumar	
Drag on Partially Contaminated Bubble Swarms in Power-Law Fluids	979
Nanda Kishore, H. Biswanath and V.S. Nalajala	

Taylor Bubble Dynamics in Pipe Fittings: A Numerical Study. 989
 Sirshendu Misra, Basanta K. Rana, Arup K. Das
 and Prasanta Kumar Das

**Evaluation of Forcing Schemes in Pseudopotential
 Based Multiphase Lattice Boltzmann Model.** 1003
 Gaddam Saritha Reddy and R. Banerjee

**Determination of Two Phase Flow Slug Velocity and Length
 Using Infrared Sensor** 1011
 J. Adhavan, C. Balachandar, S. Arunkumar and M. Venkatesan

**Implementation of Modified-CICSAM Algorithm
 for Three Dimensional Two Phase Flow.** 1019
 Bhaskar Chakraborty, Hiteshkumar Zinjala and Jyotirmay Banerjee

**Transition in the Elliptical Jet Characteristics from Super
 to Subcritical Conditions** 1029
 C.K. Muthukumaran, Kamalakannan Kannaiyan
 and Aravind Vaidyanathan

**Computational Prediction of the Performance of Positive
 Displacement Pumps** 1039
 N. Bhandari and M. Damodaran

**Spline Based Modeling of Two-Dimensional Droplets
 on Rough and Heterogeneous Surfaces.** 1049
 Ashesh Chattopadhyay, Atul Thakur and Rishi Raj

Modelling of a Single Bubble Rising in a Liquid Column 1059
 Ankit Verma, R. Babu and Malay K. Das

Flow over Fluvial Bedforms with Suction. 1069
 Mahesh Patel, Sainath Panigrahi and Bimlesh Kumar

**Performance Enhancement of Rotary Desiccant Wheel
 by Innovative Designs of Multiple Desiccant Layers** 1079
 Selvaraji Muthu, Prabal Talukdar and Sanjeev Jain

**Effect of Orifice Recess on the Droplet Size Distribution of Sprays
 Discharging from Gas-Centered Swirl Coaxial Atomizers** 1091
 Athul Joseph, P. Nandagopalan, T. John Tharakan and D. Sivakumar

**Eulerian-Eulerian Modeling of Dispersed Laminar
 Gas-Particle Flows over an Unstructured Grid.** 1101
 Subrat Kotoky, Amaresh Dalal and Ganesh Natarajan

**A Hybrid Grid Based Algebraic Volume of Fluid Method
 for Interfacial Flows** 1111
 Jai Manik, Amaresh Dalal and Ganesh Natarajan

Lattice Boltzmann Modelling of Capillarity-Induced Resonance of Blob Inside a Circular Tube 1121
 Pitambar Randive, Saurabh Bhardwaj and Amaresh Dalal

An Experimental Investigation to Analyse the Heat Transfer Characteristics of Hot Surface by Obliquely Impinging Liquid Jet . . . 1131
 Manish K. Agrawal, Mayank Modak, Palash Gupta, Satish Chandra and Santosh K. Sahu

Level Set Method Based Simulations on Impact-Dynamics of Bouncing and Non-bouncing Droplet on Super-Hydrophobic Substrates. 1141
 Nagesh D. Patil, Atul Sharma and Rajneesh Bhardwaj

Euler-Lagrange Simulations of Gas-Liquid Flow in a Basic Oxygen Furnace and Experimental Verification 1151
 Abdul Quiyoom, Vivek V. Buwa and S.K. Ajmani

Unsaturated Physical Non-equilibrium Contaminant Transport Modeling Using Modified FEMWATER. 1163
 R. Someswaran and Suresh A. Kartha

Shape and Size of the Entrained Liquid Mass by a Taylor Bubble Rising Through a Liquid-Liquid Interface 1171
 Abhimanyu Kar and Prasanta Kumar Das

Volume-of-Solid Immersed Boundary Method for Free Surface Flows with Arbitrary Moving Rigid Bodies 1181
 Jitendra Kumar Patel and Ganesh Natarajan

Motion of a Drop in Viscous Fluid Along an Inclined Plane 1193
 P. Anoop and Baburaj A. Puthenveetil

Part VII Fluid-structure Interaction and Flow-induced Noise

Experimental Investigations on Aft Ramp Cavities with Fore Wall Modifications in Scramjet Combustors 1203
 S. Jeyakumar, V. Venkateshwaran, N. Surjith, A. Karkuvel Raja and G.S. Samy

Effect of Damping Length on Dynamic Performance of Two-Stage Two-Spool Electrohydraulic Servo valve 1213
 Nitesh Mondal and Biswanath Datta

Adaptive Fuzzy—Sliding Mode Control with Fixed Bias Compensator for an Electrohydraulic Actuation System with Hard Nonlinearities 1223
 S. Chaudhuri, S. Dasmahapatra, A. Chatterjee, R. Saha, S. Mookherjee and D. Sanyal

Numerical Analysis of Flow Separation in Rocket Nozzles. 1233
 K. Sreejith, M.P. Dhrishit, M. Deepu and T. Jayachandran

Control of Cross-Wind Forces on a Finite Circular Cylinder at Subcritical Reynolds Number 1243
 Dhiraj Kumar and Kamal Poddar

Effect of Free Stream Turbulence on Flow over a Circular Cylinder in the Sub-critical Regime: An Experimental Investigation 1253
 Murugan Thangadurai, Mrityunjay Singh, Vinoth Kumar and P.K. Chatterjee

CFD Study on Flow Across a Stationary Cylinder in Between Two Counter-Rotating Control Cylinders. 1263
 Malhar Dilip Malushte and Atul Sharma

Fluid Structure Interaction of a Flexible Plate in a Compressible Medium 1273
 W. Dheelibun Remigius and Sunetra Sarkar

Effect of Attached Type Splitter Plate Length over a Square Prism in Subcritical Reynolds Number. 1283
 Manish Kumar Chauhan, Bhupendra Singh More, Sushanta Dutta and Bhupendra Kumar Gandhi

Investigation of Flow Interference Between Two Inline Square Cylinders With Different Spacing Ratios in Subcritical Reynolds Number Regime 1293
 Bhupendra Singh More, Manish Kumar Chauhan, Sushanta Dutta and Bhupendra Kumar Gandhi

Part VIII Microfluidics

Influence of Surface Contact Angle on Uphill Motion of Droplets Due to Electrostatic Actuation. 1305
 Saikat Datta, Arup K. Das and Prasanta Kumar Das

Performance Analysis of Uniform and Expanding Cross-Section Microchannels for Single Phase and Flow Boiling Heat Transfer 1315
 Yogesh K. Prajapati, Manabendra Pathak and Mohd. Kaleem Khan

Convection Inside a Pinned Water Droplet During Drying Process 1323
 T.K. Pradhan and P.K. Panigrahi

Analysis of Heat Transfer in the Slip Flow Region Between Parallel Plates	1331
Hari Mohan Kushwaha and Santosh Kumar Sahu	
Analysis of Static and Dynamic Contact Angles of Ferrofluid Droplets for Magnetically Actuated Micropumps	1341
S. Chatterjee, D. Bhowmik, A. Mukhopadhyay and R. Ganguly	
Analysis of Drop Spreading upon Impact on Dual-Textured Surface	1351
V. Vaikuntanathan and D. Sivakumar	
On-Chip Microfluidic Separation of Biological Entities in Field Flow Fractionation and Split Flow Thin Fractionation Devices	1361
Abhishek Samanta, Ranjan Ganguly, Amitava Datta and Nipu Modak	
Experimental Measurement of Vibration of Liquid Droplet at Low Bond Numbers Using ESPI	1371
Prasanna S. Gandhi, S.A. Deepak, Prashant Agrawal, Salil S. Kulkarni and Adrian Neild	
Estimation of Permeability of Porous Material Using Pore Scale LBM Simulations	1381
M. Jithin, Nimish Kumar, Malay K. Das and Ashoke De	
Part IX Bio-inspired Fluid Mechanics	
Reverse Engineering of Diffusion-Limited Controlled Drug Delivery Devices	1391
Anurag Pramanik and Sanjeev Garg	
Effect of Sinusoidal Gust on Thrust Generated by a Plunging Airfoil.	1401
Salil Harris and Sunetra Sarkar	
Particle Image Velocimetry Measurements of Rigid and Flexible Rectangular Wings Undergoing Main Flapping Motion in Hovering Flight.	1411
Srikanth Goli, Arnab Roy, Devendra Kumar Patel and Subhransu Roy	
Experimental Study of Oscillatory Flow in a Rigid and a Compliant Tubular Bulge	1421
Abdullah Y. Usmani and K. Muralidhar	
The Effects of Leading Edge Orientation on the Aerodynamic Performance of Dragon Fly Wing Section in Gliding Flight	1433
Noorul Huda and Syed Fahad Anwer	

CFD Study on Hydrodynamics of Three Fish-Like Undulating Hydrofoils in Side-by-Side Arrangement 1443
 Mukul Shrivastava, Malhar Malushte, Amit Agrawal and Atul Sharma

Part X IC Engines and Gas Turbines

Investigations on Premixed Charge Compression Ignition (PCCI) Engines: A Review 1455
 Girish E. Bhiogade, Neeraj Sunheriya and J.G. Suryawanshi

Computational Modelling of a Novel Fire Extinguisher Design 1465
 H. Sharma and M. Damodaran

Correlation of Equivalence Ratio with Spectrometric Analysis for Premixed Combustion 1475
 Uddalok Sen, Aayush Sharma, Suvabrata Panja, Saikat Mukherjee, Swarnendu Sen and Achintya Mukhopadhyay

Reacting Flow Simulation of Rocket Nozzles 1485
 S. Shyji, N. Asok Kumar, T. Jayachandran and M. Deepu

Effect of Load Level on Performance and Emission Characteristics of a Biogas Run Dual Fuel Diesel Engine 1497
 Bhaskor J. Bora, Ujjwal K. Saha, S. Chatterjee and Vijay Veer

Numerical Study of the Cavity Geometry on Supersonic Combustion with Transverse Fuel Injection 1509
 M.F. Khan, R. Yadav, Z.A. Quadri and S.F. Anwar

Performance and Emission Analysis of a Single Cylinder CI Engine using *Simarouba glauca* Biodiesel 1519
 Parashuram Bedar, P.R. Chitragar, K.V. Shivaprasad and G.N. Kumar

Part XI Transport Phenomena in Materials Processing and Manufacturing

Solar Convective Furnace for Heat Treatment of Aluminium 1531
 Deepesh Patidar, Ravindra Pardeshi, Laltu Chandra and Rajiv Shekhar

Part XII MHD and EHD Flows

Design and Characterization of a Novel Dielectric Barrier Discharge Plasma Actuator for Flow Control Application 1545
 B.K. Mishra and P.K. Panigrahi

Instability Modes and Dimensional Analysis of Micro/Nano Electro-Encapsulation Process 1555
 Kaushlendra Dubey and P.K. Panigrahi

Oscillations of a Sessile Droplet in Contact and Non-contact Modes Under an AC Electric Field	1565
Shyam Sunder	
MHD Mixed Convective Heat Transfer Over an Isothermal Circular Cylinder Using Low R_m Approximation	1575
S. Udhayakumar, T.V.S. Sekhar and R. Sivakumar	
Flow in Thermomagnetic Energy Conversion Loops	1585
Dipanjan Ray, Achintya Mukhopadhyay and Ranjan Ganguly	
A Meshfree Framework for Ideal Magnetohydrodynamics	1595
Kalpajyoti Borah, Ganesh Natarajan and Anoop K. Dass	
Part XIII Granular Flows	
Dune Formation in Horizontal Pneumatic Conveying System	1609
Atharva Patankar, Ajay Makawana and Manaswita Bose	
A Qualitative Assessment of Different Regimes During Concentric and Eccentric Discharge a Silo	1617
Ritwik Maiti, Gargi Das and Prasanta Kumar Das	
Part XIV Nuclear Reactor Thermal Hydraulics	
Steady State and Transient Characteristics of Natural Circulation Loop	1629
A.M. Vaidya, N.K. Maheshwari and P.K. Vijayan	
Entropy Generation Analysis of a Nuclear Fuel Element Surrounded by a Flow of Coolant Through an Annular Channel	1641
Rajeswar Chatterjee, Antarip Poddar, Koushik Ghosh, Achintya Mukhopadhyay and Swarnendu Sen	
Heat Transfer Characteristics of Hot Surface Using High Alcohol Surfactant by Impinging Liquid Jet	1653
Manish K. Agrawal, Mayank Modak and Santosh K. Sahu	
Numerical Investigation on an Obliquely Impinging Circular Air Jet on a Heated Flat Plate at Small Jet Plate Spacing	1663
Abhishek Bhagwat and Arunkumar Sridharan	
Numerical Investigation on Orthogonal Impingement of Circular Air Jet on a Heated Flat Plate at Low Jet Plate Spacing	1673
Bhavin Shah, Abhishek Bhagwat and Arunkumar Sridharan	
Comparison of Lumped Parameter and CFD Code Predictions: Sump Evaporation Phenomena	1685
Srinivasa Rao Ravva, Kannan N. Iyer, Aniket Gupta, Gurav Kumar, Avinash J. Gaikwad and S.K. Gupta	
Author Index	1697

About the Editors

Dr. Arun K. Saha (Ph.D., 1999, IIT Kanpur) has experience in experiments and numerical simulation in fluid mechanics and heat transfer. The numerical and experimental work done by him has helped in improving the fundamental understanding of jets and wakes. Currently he works on active control of flow and heat transfer for drag reduction of vehicles and gas turbine respectively. He is an author of several research publications and chapters in textbooks. He has been at IIT Kanpur since November 2004.

Dr. Debopam Das (Ph.D., 1998, Indian Institute of Science Bangalore) works in the area of experimental and theoretical fluid mechanics. His research work encompasses both compressible flow and incompressible flow. Compressible flow research is primarily on flow and acoustic field associated with formation of compressible vortex ring and its interaction with shock, wall, or other generic shape objects. Incompressible flow research spans across different topics; transition of unsteady internal flow, free shear and buoyancy-driven flows, unsteady aerodynamics of birds' and insects' flights. He has also designed and fabricated several flapping wing flying MAV's of different sizes including a hovering MAV. In theoretical fluid dynamics, he has contributed towards an analytical solution method for unsteady internal flow when the pressure as well as the velocity is unknown. He is an author of several research publications. He has been at IIT Kanpur since December 2001.

Dr. Rajesh Srivastava has worked extensively on numerical simulation of flow and transport through variably saturated porous media. His work provides significant insight into the transport of reactive solutes in porous media. His research finds applications in remediation of contaminated sites as well as mining of minerals through heap leaching. His present work focuses on the effect of climate change on water resources and inverse techniques for characterization of porous media and identification of possible contaminant sources. He is an author of several research publications and two text books. He has been at IIT Kanpur since December 1997.

Dr. P.K. Panigrahi is presently working as Professor and Head in the Mechanical Engineering Department, IIT Kanpur. Previously, he was the Head of the Laser Technology Program. He obtained his MS and Ph.D. from LSU, Baton Rouge, USA. His research interests include development and implementation of optical techniques for characterization of transport processes in various devices/systems at macro-scale and micro-scale involving fluid flow, heat and mass transfer.

Dr. K. Muralidhar (Ph.D., 1985, University of Delaware) has conducted a wide spectrum of experiments and numerical simulation in fluid mechanics and heat transfer. He has contributed to the understanding of transport phenomena in porous media, wave dynamics, in addition to development of numerical algorithms and laser measurement of flow and thermal fields. His research finds applications in industrial processes such as enhanced oil recovery, methane hydrates and CO₂ sequestration, regenerators, and growth of optical crystals. His present work relates to interfacial transport, biomedical imaging, and inverse techniques. He is an author of several research publications, textbooks, and monographs. He has been at IIT Kanpur since July 1987.

Part I
Fundamental Issues and Perspective
in Fluid Mechanics

Effect of Inlet Shear on the Flow Structures Associated with Elevated Jet in Cross-Flow at Low Reynolds Number

Sachidananda Behera and Arun K. Saha

Abstract A Direct numerical simulations (DNS) is carried out to study the highly three-dimensional flow field around a jet coming out of a stack and interacting with the cross-flow passing it. The major objective of the study is to assess the effect of shear strength of inlet velocity on the wake behavior and structures for elevated jets in cross-flow (EJICF). All the simulations are carried out for a Reynolds number of 250 based on the averaged inlet velocity and width of the stack and the blowing ratios considered are 0.5, 1.0 and 1.5. Effect of shear strength (0.025, 0.050 and 0.075) of inlet velocity is studied by solving the incompressible unsteady 3-D Navier-Stokes equations using a second order discretization for space and time. The solution algorithm is based on MAC method. The dynamics of flow associated with the flow field, like different wakes zones present in the flow are discussed with respect to the shear strength of free stream velocity. The vortical structure associated with EJICF is also studied.

Keywords DNS · Elevated jet in cross-flow · Vortical structures · Wake

1 Introduction

The flow field of a jet coming out of an opening normal to a cross-flow has received a large attention due to its widespread application. The jet may emit from a ground level source or from an elevated source (pipe, stack). A jet coming out from a ground level source are often encountered in practical applications, such as that of engineering and industrial interests like turbine blade cooling, fuel injection, gas turbine combustor and V/STOL aircraft. Where as an elevated jet discharging into a cross-flow arise in

Sachidananda Behera (✉) · A.K. Saha
Department of Mechanical Engineering, Indian Institute of Technology,
Kanpur 208016, India
e-mail: sbehera@iitk.ac.in

A.K. Saha
e-mail: aksaha@iitk.ac.in

many situations of environmental fields, e.g., burner, stack flare, plume dispersion and discharge of effluents. The complexities associated with the flow field of elevated jet in cross-flow are due to interactions among the jet, jet-wake, stack-wake and boundary layer developed at the wall. Flow properties of jets emitted into cross-flows from an elevated source and that from a ground level source have been studied in details in literatures, largely the attention being on the ground level case.

From environmental point of view (plume dispersion and pollutant dispersion) one of the major parameter is the velocity ratio of jet-to-crosswind and Briggs [1] advocated that this ratio mostly known as blowing ratio should always be greater than 1.5 to avoid any downwash of pollutants which could affect the near environment of the stack. But Huang and Hsieh [2] were of the opinion that to avoid down-wash the blowing ratios should be equal or greater than 2.0 and they suggested this on the basis of their experimental work. Arora and Saha [3] through their numerical study showed how the flow physics of an EJICF is affected by the blowing ratio, also the effect of blowing ratio on the concentration field was studied.

An interaction between jets coming out of a stack source with that of a cross-flow has been a major area of interest for researchers in the past, and still provides a lot of scope for further research because of the complex flow structures associated with this kind of flow. The major flow structures associated with EJICF are jet shear layer vortices, different wake zones, horse shoe vortex and counter rotating vortex pair (CRVP). Two types of wakes are discussed in literatures in relation to EJICF, namely stack wake and jet wake [4]. As described by Moussa et al. [5] the jet wake is developed because of the shedding of vortices behind the jets, as the vertically ejecting jet itself behaves as a bluff body. The stack wake get developed because the stack behaves as an obstacle to the cross-flow and the vortex shedding pattern is very similar to that observed in case of bluff bodies [4].

Eiff et al. [4] advocated about the presence of two kinds of wakes in reference to EJICF and showed how vortices associated with the jet wake interact with wake associated with stack. To this point no work is reported on the effect of inlet velocity profile on the wakes associated with EJICF and therefore the present work intends to study the effect of shear strength of inlet free stream velocity on the flow structures.

2 Solution Methodology

The 3-D unsteady Navier-Stokes equations for incompressible flow were solved using finite difference method to study the interaction of a square jet and cross-flow. The governing equations used are all in non-dimensional form. To achieve the non-dimensional form of the governing equations, the velocities are non dimensionalised with respect to the mean velocity of cross-flow, u_m , and all length scales with respect to width (D) of the square stack. The pressure is non dimensionalised with respect to ρu_m^2 and time with D/u_m . The governing equations are solved on a non-uniform staggered grid arrangement such that the velocities are defined on

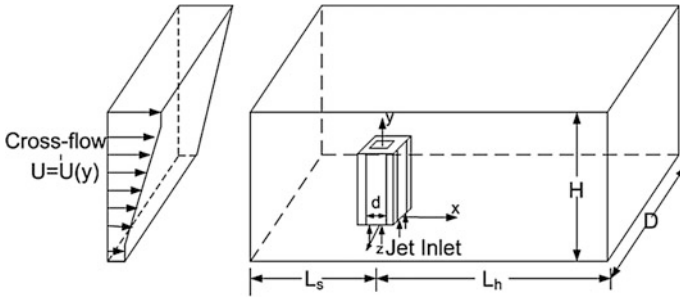


Fig. 1 Schematic of the computational domain

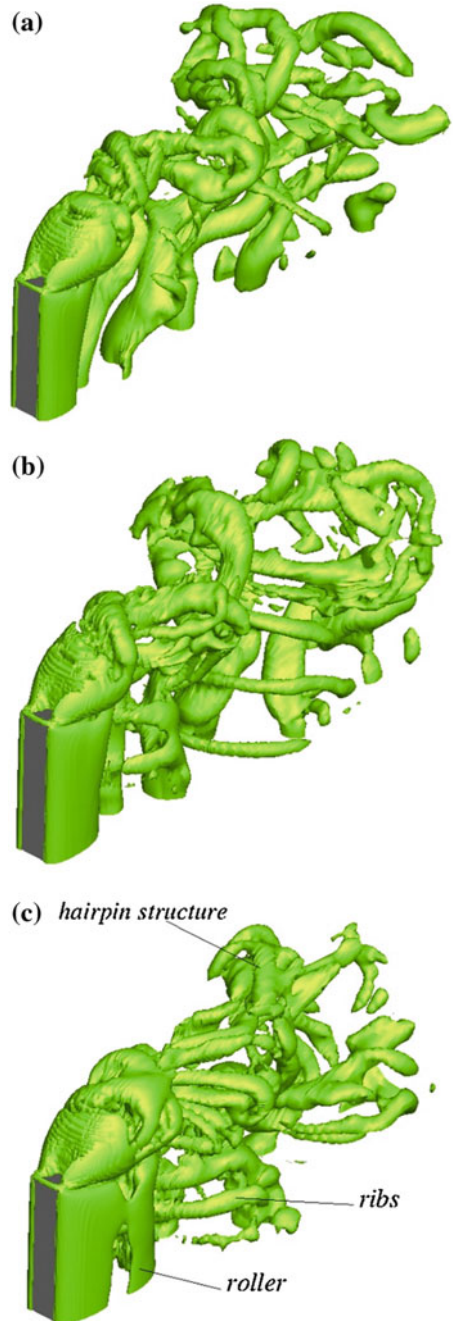
faces of the cell to which velocity is normal and pressure is defined at the center of the cells. An explicit second order in time, Adams-Bashforth differencing scheme (second order accurate in time) has been used for time advancement of the advection and diffusion terms for each case. Second order central differencing scheme is used for the spatial discretization of both the advection and diffusion term. The boundary conditions used for the present investigation are: (i) at the cross flow inlet, a shear velocity profile of different shear, with its mean being unity is used and the vertical as well as cross stream velocity is considered as zero. (ii) Since our main aim is to study the effect of shear on jet wake and stack wake, we considered a certain portion of the chimney instead of the whole length and applied free slip boundary condition near the bottom of the computational domain except for the zone bounded by the stack. (iii) Free slip boundary condition are imposed on the transverse confining surfaces as well as on the top confining surface. (iv) No-slip boundary conditions are imposed on the walls of the stack. (v) A convective boundary condition as suggested by Orlanski [6] is used as the outlet boundary condition at the exit of the computational domain. For all the computations carried out in the present work the grid size is chosen to be $222 \times 114 \times 148$. The details of the code validation and grid independence test were reported elsewhere [3]. A schematic of the complete computational domain is shown in Fig. 1.

3 Results and Discussion

3.1 Instantaneous Vortical Structures Associated with the Flow Field

To capture the various coherent structures associated with the flow “Q-criterion” technique given by Dubief and Delcayre [7] is used in the present study. Figure 2 presents the vortical structures at $BR = 1.0$ for $K = 0.025, 0.05$ and 0.075 . The vortical structures near to the stack region and the exit region of the jet are clearly brought into visualization by the iso-surface plots of the vortical structures. The

Fig. 2 Instantaneous vortical structures ($Q = 0.25$) at $BR = 1.0$ for various shears: **a** $K = 0.075$, **b** $K = 0.05$ and **c** $K = 0.025$



plots of vortical structures clearly shows the presence of hairpin structures, rollers type structures, ribs type structures and stack shear layer. Similar structures associated with EJICF are earlier reported by Arora and Saha [3]. The formation of the roller type structures can be credited to the separating shear layer in the wake of the stack that forms into these structures. However far downstream these rollers are found to be absent, which may be due to the diffusion of vortices with increasing downstream direction. Similar coherent structures (rollers and ribs) are also found for bluff body wakes. A closer look at the structures near the stack reveals the presence of more rib structures in the stack wake for $K = 0.025$, where as the number of rib like structures is minimum for $K = 0.075$. The presence of more rib structure in case of $K = 0.025$ may be due to the interaction between the up-shooting jet and strong downwash present in the flow as compared to the other two cases. The absence of rib structures in case of $K = 0.075$ can be related to the weaker downwash present in the flow field.

A closer look at the jet exit zone shows the presence of hairpin structures in the wake region of jet and the formation of these hairpin structures are mainly due to the interaction between the cross-flow and the jet coming out of the stack. These hairpin structures are first developed and then shed in the down-stream direction. It is also noted that as the hairpin structures convect down stream they get twisted, mainly because of the shear present in the flow as compared to a flow without shear Arora and Saha [3].

3.2 *Instantaneous Flow Field*

The instantaneous vorticity contour (ω_z) on the mid-xy plane are shown in Fig. 3. It is evident from Fig. 3 that considerable amount of vorticity is present at the inner walls of the stack. The vorticity generated inside the stack is convected out of the stack by the up shooting jet. Once it comes out of the stack it gets convected down-stream by the action of crossflow. The instantaneous contour of vorticity clearly shows the presence of Kelvin-Helmholtz kind of instabilities. It is also observed from the vorticity contour plot that the immediate near region (along the span of stack) is most affected in case of shear $K = 0.025$ and least affected in case of shear $K = 0.075$. This is mainly because of low local blowing ratio in the former case and high local blowing ratio in the latter case.

3.3 *Time Averaged Flow Field*

The time averaged streamlines on the $Z = 0$ plane are shown in Fig. 3. To get the average flow field data, all the scalar and vector quantities of the flow field are time averaged over a longer duration of time i.e. around 60 non-dimensional times. The bifurcation lines present in all the three cases are found to be inclined at an angle to